

## Autodesk Inventor 2018 Design Variations Representations

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### Autodesk Inventor 2018 Design Variations

The Autodesk® Inventor® 2018: Design Variations and Representations student guide contains topics that teach how to efficiently create and represent designs based on existing geometry. You will learn how the iFeature, iPart, and iAssembly tools can leverage existing geometry to quickly create additional or slightly varied geometry.

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Factory Design Suite 2018, Inventor 2018, & Product Design Suite 2018. By: Help . Help. 0 contributions. In-Product View . ... fitting variations, and shaft lengths. ... Get answers fast from Autodesk support staff and product experts in the forums. Visit Inventor forum.

## **iAssemblies | Inventor 2018 | Autodesk Knowledge Network**

In this streamlined workflow, you select a component, save a copy of that component to disk, and replace the selected component within the assembly with the newly created copy while maintaining all assembly constraints. Tip: Use this command to investigate design variations. On the ribbon, click Assemble tab Productivity panel Save and Replace . Click to select a component. Attention: You can ...

## **Save and Replace | Inventor 2018 | Autodesk Knowledge Network**

In part authoring, you design the part and define all of its variations. Start with a new or existing part or sheet metal part. Determine the portion of your design that changes with each

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member. Rename parameters, establish equations, and create user parameters. Use the Create iPart command to define table rows that represent versions (members).

## **About iParts | Inventor 2018 | Autodesk Knowledge Network**

Stress Analysis settings define the defaults for all new simulations. You cannot make changes to the settings while working in a simulation. Changes to the settings apply only to simulations that you create afterward. Create a simulation Open the model you want to analyze. By default, it opens in the modeling environment. On the ribbon Environmentstab Begin panel, click Stress Analysis .

## **Create a simulation | Inventor 2018 | Autodesk Knowledge ...**

Autodesk Inventor Content Center libraries provide standard parts (fasteners, steel shapes, shaft parts) and features to insert in assemblies. Two types of parts are included in the Content Center library: standard parts and custom parts. Standard parts (fasteners, shaft parts) have all part parameters defined as exact values in the table of parameters.

## **Content Center parts | Inventor 2018 | Autodesk Knowledge ...**

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geometry. You will learn how the iFeature, iPart, and iAssembly tools can leverage existing geometry to ...

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The iLogic component of Inventor software is a powerful tool that is utilized in automating the design process, and improving efficiency and the quality of your work. In this class, we will discuss and demonstrate how powerful the iLogic tool is in letting you automate your design workflows. We will start our discussion with how we used iLogic when we started to learn how to apply it to our ...

## **The Power of iLogic Design ... - Autodesk University**

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